ARBITRARY LAGRANGIAN-EULERIAN FORM OF FLOWFIELD DEPENDENT VARIATION (ALE-FDV) METHOD FOR MOVING BOUNDARY PROBLEMS

BY

MOHD FADHLI BIN ZULKAFLI

A thesis submitted in fulfilment of the requirement for the degree of Doctor of Philosophy

Kulliyyah of Engineering
International Islamic University Malaysia

JUNE 2015

ABSTRACT

Flowfield Dependent Variation (FDV) method is a mixed explicit-implicit numerical scheme that was originally developed to solve complex flow problems through the use of so-called implicitness parameters. These parameters determine the implicitness of FDV method by evaluating local gradients of physical flow parameters, hence vary across the computational domain. The method has been used successfully in solving wide range of flow problems. However it has only been applied to problems where the objects or obstacles are static relative to the flow. Since FDV method has been proved to be able to solve many complex flow problems, there is a need to extend FDV method into the application of moving boundary problems where an object experiences motion and deformation in the flow. With the main objective to develop a robust numerical scheme that is applicable for wide range of flow problems involving moving boundaries, in this study, FDV method was combined with a body interpolation technique called Arbitrary Lagrangian-Eulerian (ALE) method. The ALE method is a technique that combines Lagrangian and Eulerian descriptions of a continuum in one numerical scheme, which then enables a computational mesh to follow the moving structures in an arbitrary movement while the fluid is still seen in a Eulerian manner. The new scheme, which is named as ALE-FDV method, is formulated using finite volume method in order to give flexibility in dealing with complicated geometries and freedom of choice of either structured or unstructured mesh. The method is found to be conditionally stable because its stability is dependent on the FDV parameters. The formulation yields a sparse matrix that can be solved by using any iterative algorithm. Several benchmark stationary and moving body problems in one, two and three-dimensional inviscid and viscous flows have been selected to validate the method. Good agreement with available experimental and numerical results from the published literature has been obtained. This shows that the ALE-FDV has great potential for solving a wide range of complex flow problems involving moving bodies.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

Roman symbols

$\boldsymbol{a}$	Convection Jacobian
$\boldsymbol{b}$	Diffusion Jacobian
$\boldsymbol{c}$	Diffusion gradient Jacobian
E	Total energy per unit mass
$\boldsymbol{F}$	Inviscid flux
$\boldsymbol{G}$	Viscous flux
I	Vertex/ cell element index number
J	Neighbour vertex/ cell element index number
M	Mach number
$\boldsymbol{n}$	Normal vector
p	Pressure
Pr	Prandtl number
Re	Reynolds number
s_a	First-order implicitness parameter
s_b	Second-order implicitness parameter
s_1	First-order convection FDV parameter
s_2	Second-order convection FDV parameter
s_3	First-order diffusion FDV parameter
s_4	Second-order diffusion FDV parameter
t	Time
u_i	Velocity elements in Cartesian coordinates

$\boldsymbol{U}$	Conservative variables
$\boldsymbol{v}_m$	Mesh velocity
x_i	Cartesian coordinates

Greek symbols

Δ	Changes in time/ space
δ_{ij}	Kronecker delta
ε	Internal energy per unit mass
Γ	Control surface/ cell boundary
γ	Specific heat ratio (Ideal gas = 1.4)
μ	Dynamic viscosity coefficient
ρ	Density
τ_{ij}	Viscous stress tensor
Ω	Control volume/ cell volume

Subscripts

I	Cell indices
$I(J)$	Neighbour cell of cell I
i, j, k	Spatial direction indices
IC	Interior cell
m	mesh
n	Normal direction
T	Total value
t	Tangential direction

w Wall surface

∞ Free stream condition

Superscript

m GMRES iteration

n time level

LIST OF ABBREVIATIONS

ALE	Arbitrary Lagrangian-Eulerian
ALE-FDV	Arbitrary Lagrangian-Eulerian form of Flowfield Dependent Variation
AoA	Angle of Attack
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CBS	Character-based Split
DGCL	Discrete Geometric Conservation Law
DGFEM	Discontinuous Galerkin Finite Element Method
et al.	*(et alia)*: and others
FDV	Flowfield Dependent Variation
FDM	Finite Difference Method
FEM	Finite Element Method
FVM	Finite Volume Method
GCL	Geometric Conservation Law
GMRES	General Minimal Residual
GRAFSS	General Relativistic Astrophysical Flow And Shock Solver
HOC	Higher-Order Compact
HOC-FDV	Higher-Order Compact Flowfield-Dependent Variation
IB	Immersed Boundary
i.e.	*(id est)*: that is
LSGR	Least Square Gradient Reconstruction
MFDV	Modified Flowfield Dependent Variation
MUSCL	Monotone Upwind Schemes for Conservation Law
PDE	Partial Differential Equation
PISO	Pressure Implicit with Splitting of Operators
SIMPLE	Semi-implicit Method for Pressure-Linked Equations
TVD	Total Variation Diminishing

CHAPTER ONE

INTRODUCTION

1.1 INTRODUCTION

Governing equations in fluid mechanics are a coupled system of nonlinear partial differential equations, which are difficult to solve analytically and to date except for some particular problems, there is no general closed-form solution to these equations. Thus, numerical approach is important in order to study and analyze the problems involving fluids. In fluid mechanics, the area that studies such an approach is Computational Fluid Dynamics (CFD), and its development began with the advent of the computer in the 1950s. CFD is important as a research and design tool today because the development of modern technologies such as high-speed transportation, electronics and biotechnologies also rely on the understanding of fluid mechanics.

Major basic techniques used in the solution of partial differential equations in general and CFD in particular are Finite Difference Methods (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). FDM is easy to formulate but because a structured mesh is required, it has difficulties with multi-dimensional problems that involve complex geometries. In contrast, complex geometries and unstructured meshes are easily accommodated by FEM but it uses large computer memory, thus slow for large problems and not well suited for turbulent flows.

FVM however, has an advantage in memory usage and speed for very large problems. This method is based on the discretization of the integral form of nonlinear partial differential equations (PDE) into finite control volumes and control surfaces. It is not limited to simple meshes as the finite volumes could take arbitrary shapes, thus applicable to unstructured grids and complex geometries. Furthermore, the system of algebraic equations by finite volume methods enforces the conservation of all variables across the control surfaces. Therefore, the conservation of mass, momentum and energy are assured in the formulation itself while variables may not be continuously differentiable across shock or other discontinuities, which is an advantage for high-speed flow problems.

In CFD, the problems are usually solved by different techniques depending on the physical properties of the flows. For example, incompressible flows are analyzed using the pressure-based formulation but compressible flows are analyzed using the density-based formulation. In dealing with the domains, which contains flows of all speed with various physical properties, where the equations of state for compressible and incompressible flows are different, and where the transitions between laminar and turbulent are involved, very special and powerful numerical treatments are needed. The so-called Flowfield Dependent Variation (FDV) theory, which was first introduced by Chung (1999), has been devised toward resolving these issues. The theory introduced the so-called FDV parameters, which are dependent on the gradient of changes between flow variables (e.g. Mach number or Reynolds numbers) of local adjacent nodal points in the computational domain. Because of these parameters, the terms containing the fluctuation variables in the FDV equation automatically follow the current physical phenomena and adequate numerical controls (artificial viscosity) are automatically activated according to the current flow field physics. The numerical scheme of the FDV equation itself will then adjust accordingly for every node based on the flow properties of different regions that coexist in the computational domain.

For the numerical simulation and analysis of objects that move within the flow, the objects usually are made static relative to the flow in the computational domain, similar to the wind tunnel experiment. However, there are many situations when the objects are needed to move or deform in the computational domain such as airfoil oscillations, wing flutter, and rotating propellers problems. This is one of the important issues in CFD applications, because the simulations of the flow around moving objects require special interpolation methods to handle the moving boundaries. Methods for moving computational mesh have been studied actively by the CFD community because of their engineering importance. One of the most popular techniques in solving moving boundaries problems is Arbitrary Lagrangian-Eulerian (ALE), which combines Lagrangian and Eulerian description of a continuum, i.e. fluid and solid, in one numerical scheme.

The present research studies the combination of FDV method and ALE method in finite volume form. The finite volume form would make this method applicable to complicated geometries of moving bodies and by combining FDV with ALE method, it would give an accurate prediction of the interactions between fluid and the moving bodies. Therefore, it is expected that the proposed method will provide a new technique of resolving accurately the interaction of arbitrary bodies in arbitrary flow fields.

1.2 PROBLEM STATEMENT

Unified Computational Fluid Dynamics (CFD) method has been the aim of the CFD community in recent years. The need for a unified method arises because in CFD, different type of flow problems need different type of method to solve, but in reality different type of flow do exist in the same region. For example, low speed flow area may coexists with high speed flow area such as in the cases of aircraft landing or take-off configurations where the free stream Mach number is much less than the local Mach number around the high-lift devices of the aircraft. Flowfield Dependent Variation (FDV) method has been introduced to resolve these problems, however this method is currently limited to stationary bodies and has not yet been used to handle moving boundary problems. This research proposes to combine FDV method with moving boundary interpolation technique, ALE method for solving moving boundary problems because in some cases, deformation and motion of the bodies need to be taken into account in order to get accurate results without ignoring its physical properties as well as the existence of many different flow regimes within a flow field.

1.3 RESEARCH PHILOSOPHY

The philosophy of this research is to combine the advantages of FDV theory with a moving body technique, the ALE method, in order to develop a robust and versatile method, which could be used for the computation of flow fields with moving boundaries. The philosophy is driven by the need to consider the deformation and/or movement of bodies in a flow in which to date, the FDV theory has not been applied. The philosophy is based on obtaining the parameters of the FDV equations from the current flow field variables at each time step and every grid point which are used to adjust governing equations in each flow region according to the current flow field situation. The combination of ALE and FDV method will be used to handle the problems involving moving boundaries in a flow. Meanwhile, the finite volume method gives the ALE-FDV formulation, the capability to solve flow problems involving bodies with complicated geometries.

1.4 RESEARCH OBJECTIVES

The objectives of this research are as follows.

- To develop a technique which could be used for wide ranges of compressible-incompressible, viscous-inviscid, laminar-turbulent, and high-low speed flows for moving boundary problems.
- To combine FDV method and a moving body interpolation technique, named ALE method for solving flow problems involving moving boundaries.
- To apply the proposed method to solve three-dimensional inviscid and viscous flow problems involving moving boundaries by developing an algorithm and translate it into an efficient computer code.
- To investigate such combinations that will satisfy the stability requirement as well as guarantee accuracy and efficiency.

1.5 RESEARCH METHODOLOGY

The methodology of this research will focus on the development of the proposed method based on numerical work. The numerical work will be carried out using traditional way of CFD, starting with pre-processing, then solving process, and ending with post-processing. Pre- and post-processing will be performed using commercial softwares, Gambit as mesh generation software and Paraview as visualization software. The solving process will be carried out using FORTRAN code on a UNIX based machine. The steps of algorithm development for the proposed method will be done as follows:

a) The step begins by expanding conservative variables (i.e. density, momentum and energy) with respect to time in a special form of a Taylor series. This expansion is done with the addition of implicitness parameters into the first and second time derivatives of that series. Next, the time derivatives are changed into spatial derivatives by substituting the Navier-Stokes equations in that series to construct the FDV formulation.

b) The general formulation of ALE-FDV method is derived by combining ALE technique with the FDV formulation. The formulation is then discretized using appropriate finite volume method.

c) Strategies to solve the ALE-FDV formulation are then constructed and translated into computing algorithm. This algorithm is then written in FORTRAN language as the solver code.

d) Then, selected stationary body in one-dimensional problems flow is solved using the developed solver and numerical as well as experimental data available in the literature is used to validate the solver.

e) If the solutions of the stationary body problems are valid, several one-dimensional moving boundary problems are selected and solved using the complete solver for the validation and analysis process.

f) Finally, ALE-FDV method is applied to two and three-dimensional flow problems by repeating steps b to e.

To summarize, the following flow chart shows the overall methodology of this research.

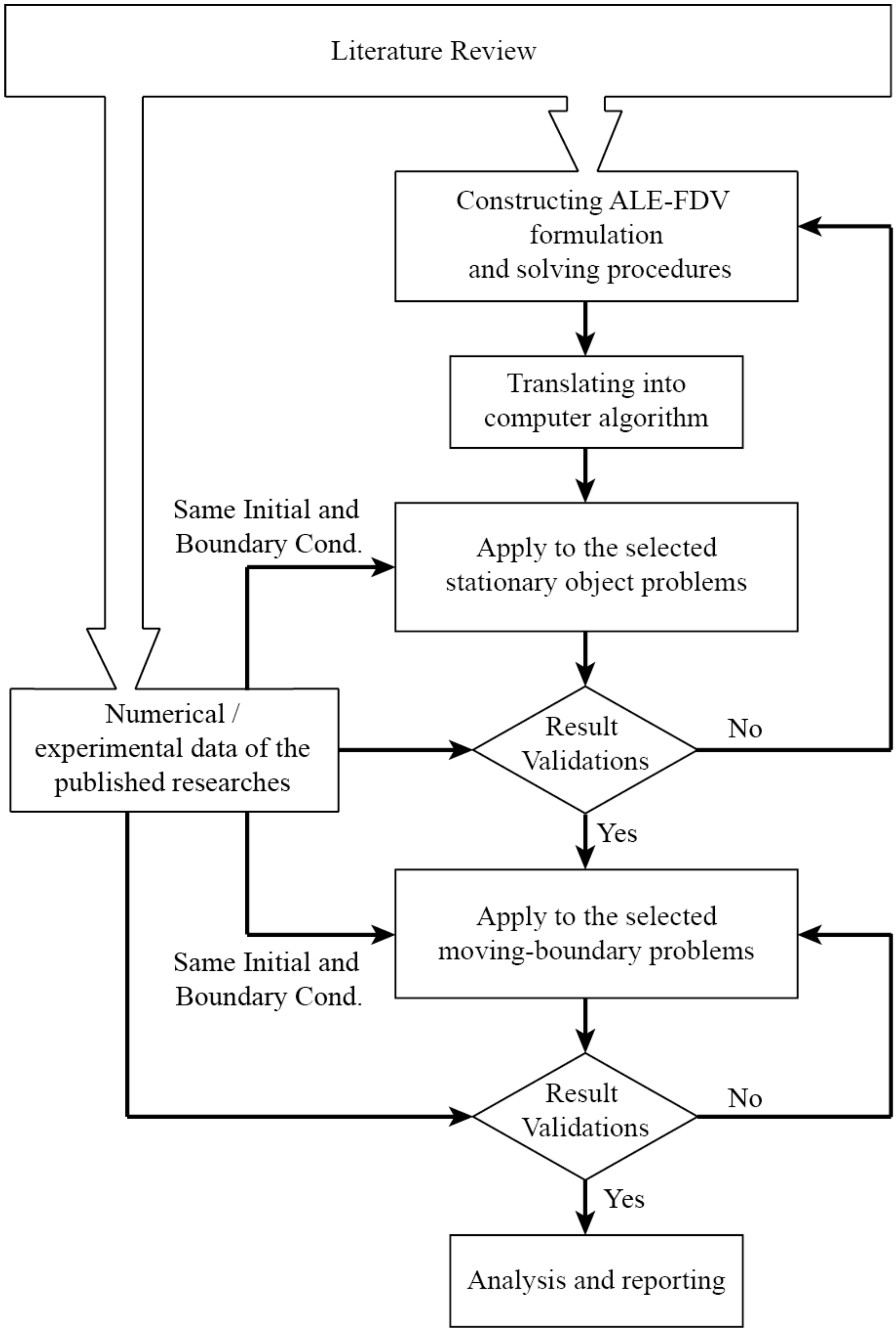

Figure 1.1 Flow chart of research methodology

1.6 SCOPE OF RESEARCH

This study focuses on developing a new numerical scheme that combines the FDV method and ALE method. The proposed method is spatially discretized using appropriate finite volume method. A flow solver, called as ALE-FDV solver is developed by translating the discrete formulation into an algorithm. In order to validate the scheme, the developed solver is applied to several benchmark one, two and three-dimensional flow problems. The flow problems involving moving mesh and fluid-structure interaction in inviscid or viscous fluid are also in the scope of this work.

1.7 THESIS ORGANIZATION

This thesis is organized by dividing it into five chapters. This chapter, which is the first chapter, gives an introduction and background of FDV and ALE method as well as demonstrates the importance of extending FDV method into the application of moving boundaries and fluid-structure interaction problems. Problem statements, research philosophy, objectives, methodology and scope of the research are also explained in the first chapter. Second chapter presents a review of previous studies and works that are relevant to this research. The review covers past and recent works related to the FDV and ALE methods as well as other works involving moving boundaries and fluid-structure interaction applications.

The third chapter explains the derivation of the ALE-FDV method and the technique to discretize it with finite volume method. The strategies to apply ALE-FDV method and the algorithm used in this research are also explained in detail in the third chapter. The fourth chapter demonstrates the applicability of ALE-FDV method in solving various flow problems involving moving boundaries. Discussion on the numerical results and verification of ALE-FDV method by comparing with previous relevant works are discussed in the same chapter. Finally, the fifth chapter concludes this research works and findings, highlights the main contributions of the research and recommends future works on improving the ALE-FDV method.

CHAPTER TWO

LITERATURE REVIEW

2.1 INTRODUCTION

Interaction between a solid and a fluid is a common phenomenon either in nature or induced by human activities. Such interactions happen in various disciplines and at different scales, which can be modeled through experiments or simulated using computational method. In a modern world, computer simulation has become an important tool because it provides an economical approach as well as additional insight to the analysis of such fluid interactions.

The numerical approach in fluid mechanics, Computational Fluid Dynamic (CFD), has been developed in the 1950s and since then has been extended by researchers in order to allow investigation on various complex fluid interactions. CFD is being used extensively in many industrial sectors and is advancing rapidly as more complex fluid interaction become available and simulation on larger scale is more in demand today.

As the flow simulation become larger and complex, development of more accurate and efficient numerical scheme has become significant. One of the numerical approaches called Flowfield Dependent Variation (FDV) method has been developed towards resolving complex flow interaction problems. In this chapter, a review on the development and application of FDV method and other similar method will be presented. At the same time, many fluid and solid interaction problems require computation of structural movement, hence require special grid interpolation technique for deformable mesh to produce accurate solutions. This chapter also presents an overview on the development and application of such grid interpolation techniques. This chapter will be ended by a summary of all reviews and conclusion on the advantages and drawbacks of numerical schemes used in the reviewed literatures.

2.2 UNIFIED CFD METHOD

Compressible flow field solvers use density as their primary unknown variable while artificial compressibility method (Chorin, 1967) and pressure correction method (Harlow and Welch, 1965) are the two approaches that have been widely used as incompressible flow solvers. The problem with compressible flow solver is that some areas of the flow would be incompressible, thus make it unsuitable for the compressible flow solver alone to compute the entire flow field. Since the solution of incompressible flow can be obtained as part of the compressible flow formulation, the method of extending the function of compressible flow solver has been actively studied (Wesseling, 2000).

Either explicit or implicit time stepping scheme can be used to solve the governing equations numerically. However, an explicit scheme needs to satisfy the stability condition in order to produce solutions of the problems. In particular, the stability condition, known as Courant-Friedrich-Lewy (CFL) condition dictates the Courant number (i.e., ratio of physical propagation speed to numerical propagation speed) must be less than unity (Courant, Friedrichs and Lewy, 1967). In other words, the time needed for the numerical information to propagate in a spatial distance Δx must be smaller than the time of physical information propagation in that same distance. Moreover, the time needed for the numerical information to propagate will become much smaller if acoustic effects are present in low-subsonic flow. When this happens, a system of governing equation will become stiff, i.e. this system requires many small steps to be solved. Moreover, often-wrong results are obtained when a stiff system of equations are solved by carrying out very large number of small time steps as reported by Turkel, Fiterman and Van Leer (1993) and Guillard and Viozat (1999).

In order to resolve the problem, some numerical schemes such as preconditioning techniques are developed. This scheme alleviates the problem by modifying the governing equations artificially by multiplication of the time-derivative with a specific preconditioning matrix. While the scheme resolved stiffness and accuracy problem for stationary solutions, the time accuracy is lost due to the modification of time derivatives in the governing equations. Therefore, to compute unsteady flows, the preconditioning method has been combined with dual time stepping method in order to restore the accuracy in time (Weiss and Smith, 1995). It has been shown that dual time stepping is more efficient than physical time stepping used by the original compressible flow solver. However, because of the large number of pseudo-time steps required for each physical time step, the efficiency lags behind incompressible flow solvers (Wesseling, 2000). As remarked by Paillére, Clerc, Viozat, Toumi, and Magnaud (1998), implicit time stepping scheme must be used if the explicit time stepping scheme requires a very small time step to satisfy the stability condition.

A different approach has been taken by Yoon and Chung (1996), where they introduced the so-called mixed explicit implicit generalized Galerkin spectral element method (MEI-GG-SEM). Unlike traditional methods, the so-called flow field dependent parameters detect the physical properties of the fluid and then automatically apply adequate computational requirements for compressible and incompressible flows. They aimed for the direct numerical simulation of turbulence flow where mesh refinement were carried out adaptively until shock waves were resolved in which the traditional turbulence model is no longer needed.

Complexities of fluid phenomena are not only due to the co-existence of compressible and incompressible flow, it also includes transition to turbulence, relaminarization, flow separation and transition between viscous and inviscid flow regions. Instead of focusing only on the incompressible and compressible flow mixture problems, a new approach called the flow field dependent mixed explicit-implicit (FDMEI) method has been developed in attempt to resolve the complexities of fluid phenomena in all speed flow regimes (Chung, 1997a, 1997b; Yoon, Moon, Garcia, Heard, and Chung, 1998). Based on the flowfield dependent variation (FDV) theory, this method uses FDV parameters which depend on the change of either Mach numbers, Reynolds numbers, Peclet numbers, or Damkohler numbers at adjacent nodal points to detect physical properties of each nodal points. Peclet number or Damkohler number is used for high speed compressible flow problems such as in hypersonic flow or chemically reactive flow problems such as in combustion. Peclet number is defined as the ratio of convective to diffusive strength (Wesseling, 2000) while Damkohler number defined the relationship between chemical reaction and flowfield transport phenomena such as convection, diffusion or heat conduction (Chung, 2002).

Appropriate numerical schemes are provided to each nodal point if necessary by calculating and updating the values of these parameters at each time step. Yoon et al. (1998) used FDMEI method to compute flow over a flat plate, supersonic flow on a compression corner, three-dimensional duct flow, and lid-driven cavity flow. Their results showed good agreement with experimental and other published numerical results. Wide range of validation results proved the capability of FDV theory to resolve mutual interactions and transition between viscous/ inviscid, compressible/ incompressible, and laminar/ turbulent flows.

2.3 FLOWFIELD DEPENDENT VARIATION (FDV) METHOD

The original idea of FDV theory began from the need to address the physics involved in shock wave turbulent boundary layer interactions (Chung, 1999). In this situation, transition and interactions of inviscid/ viscous, compressible/ incompressible, and laminar/ turbulent flows constitute not only the physical complexities but also computational difficulties. This is where the very low velocity in the vicinity of the wall and very high velocity far away from the wall co-exist within the domain of study. Implicitness parameters were initially introduced in the expansion of conservative variables in the Taylor series up to the second order time derivatives. The series are then used in the Navier-Stokes system of equations so that these parameters could be used in solving the flow field interaction. These parameters are characterized into two categories, namely first and second order convection/diffusion parameter. In particular, the first order parameters ensure the solution accuracy and second order parameters assist in the solution stability, and both serve as physical parameters to allow the transitions and interactions of different types of flow to be automatically accommodated.

Moreover, flowfield-dependent variation formulations have been addressed by Schunk, Canabal, Heard, and Chung (1999) as a strategy toward unification of finite difference, finite element, and finite volume methods. All the physical phenomena are taken into account in FDV equations, so that spatial discretization will not dictate the physics, but rather are no more than simply the options on how to discretize between adjacent nodal points or within an element. On the other hand, the FDV parameters introduced in FDV equations play significant roles as adjusting the governing equations (hyperbolic, parabolic, and/or elliptic), resolving various physical phenomena, and controlling the accuracy and stability of the numerical solution. The theory is verified by a number of example problems addressing the physical implications of the variation parameters, which resemble the flow field itself. Using finite difference method as spatial discretization of FDV equations, Schunk et al. (1999) showed numerical results of three-dimensional triple shock/boundary layer interaction matched with experimental results and finite difference calculation using k-ε model as reported by Garrison, Settles, and Hortsman (1996), thus indicated that FDV theory was robust enough to adequately model complex flow phenomena.

FDV theory also has been applied in high-energy astrophysics problems, particularly to those containing shock waves and high-speed flow. Richardson, Chung, Karr, and Pendleton (2000) proposed the FDV theory as a method to accurately solve very high-speed flow problems and capturing relativistic shocks. Instead of Mach number, Lorentz factor, which describe the velocity of an object relative to the speed of light (Corcoran, 2010), is used to dictate the FDV convection parameters. The theory has been applied in relativistic hydrodynamic equations to solve relativistic shock tube problems. Furthermore, they also presented FDV method for solving general relativistic non-ideal hydrodynamics (Richardson and Chung, 2002a). Non-ideal flows are where radiation, magnetic forces, viscosities, and turbulence play an important role. Relativistic effects become pronounced in such cases as jet formation from black hole magnetized accretion disks, which may lead to the study of gamma-ray bursts. Richardson and Chung (2002b) implemented the FDV theory to obtain general relativistic astrophysical flow and shock solver (GRAFSS) which is a multi-dimensional finite element code based on the FDV theory capable of solving complex geometries. Richardson, Cassibry, Chung, and Wu (2010) have demonstrated the capability of the finite element form of FDV method in physical applications that have widely varying spatial and temporal scales. The use of a finite element formulation also adds capabilities such as flexible grid geometries and exact enforcement of Neumann boundary conditions. The author presented the results of converging/diverging nozzle, which contains both incompressible and compressible flow in the flow field over a range of subsonic and supersonic regions. The results showed that the finite element formulation is stable and accurate for a range of both Mach numbers and Lorentz factors while its accuracy are comparable to other methods and slightly better than Total Variation Diminishing (TVD) method.

Heard (2007) utilized the FDV parameters for adaptive mesh refinements. FDV equations were solved using an element-by-element GMRES solver with the elements grouped together to allow the element operations to be performed in parallel. Besides dictating the physical properties of the flow field, FDV parameters are used as error indicators for a solution-adaptive mesh. The finite element grid is refined as dictated by the magnitude of FDV parameters. This method is comparable to those where the grid is refined using primitive variable error indicators, and requires less computational time to generate the grids. The use of parallel processing in performing some element operations is shown to reduce the wall clock time by approximately 40 percent in going from one to eight processors. The algorithm's ability to solve a flow field containing various kinds of interactions is demonstrated by solving a variety of fluid flow conditions ranging from low-speed incompressible flow to compressible flow containing shock waves, and the refinement of finite element grid to further resolve discontinuities in the flow field have been successful.

FDV parameters were modified by Megahed, El-Mallah, and Girgise (2006) to improve the understanding of physical meaning of the variation parameters. The modified method, MFDV method was applied to the Euler equation with standard Galerkin finite element method as its spatial discretization. Two well-known cases of supersonic internal flow; shock reflection problem and compression corner problem were solved and the results were shown to have a good agreement with other published literature. Furthermore, three other cases of supersonic internal flow; half wedge in supersonic wind tunnel, extended compression corner problem, and circular arc problem also has been solved. All of the numerical solutions are comparable with analytical and numerical solutions obtained by other established methods, thus showed the ability of the MFDV method to solve problems involving supersonic internal flow.

FDV theory also has been combined with higher-order compact method (Hirsh, 1975; Lele, 1992; Mawlood, Basri, Asrar, Omar, Mokhtar, and Ahmad, 2006; Elfaghi, Asrar, and Omar, 2010) which is generally a technique that use fewer number of nodal points to obtained high order finite difference approximation as opposed to classical finite difference method. Originally developed by Elfaghi et al. (2010), Higher-Order Compact Flowfield-Dependent Variation (HOC-FDV) method used implicit fourth order compact differencing Hermitian (Pade-type) scheme to approximate the spatial derivatives in FDV equations. HOC-FDV method has been applied to solve up to two-dimensional problems such as Sod-shock tube problem, interaction of two-blast shock waves, and flow past NACA0012 airfoil (Elfaghi, Asrar, and Omar, 2009a). The same method also has been used in solving full Navier-Stokes equations such as nonlinear viscous Burgers equation and transient Couette flow (Elfaghi, Asrar, and Omar, 2009b, 2010). From the literatures, HOC-FDV method has shown the ability to operate in various flow regimes without using any special treatments due to the capability of the FDV method. Moreover, HOC-FDV is more efficient than FDV method as well as other conventional high-order methods because less stencil points is required to obtain solutions with the same accuracy.

2.4 FLOWS WITH MOVING BOUNDARIES

Moving boundary problems in CFD applications are the cases where boundaries of the bodies (or obstacles) in a flow are moving and/or deforming. For example, airfoil oscillations, wing flutter, accelerated/decelerated aircraft, rotating propellers, fast turning cars, reciprocating engines, suspension bridges vibration, flapping wings, pulsating blood vessels, etc. Some movements of these boundaries are relatively small but when they undergo large displacements, rotations or deformations, the effects of fluid-body (or fluid-structure) could not be ignored. The need to solve such kind of flow problems using dynamic mesh (i.e. the moveable/deformable body-conformal grids system) has attracted many researches to develop various kinds of moving grid interpolation techniques. One of the body-conformal moving grid interpolation technique that is widely used in fluid and solid mechanics is Arbitrary Lagrangian-Eulerian (ALE) method. However, simulation of largely deformable objects using dynamic mesh is quite unstable and requires costly grid generation methods. Therefore, such simulations are widely performed using stationary mesh (non-conformal grid system) with the so-called Immersed Boundary (IB) method (Peskin, 1977; Mittal and Iaccarino, 2005).

2.4.1 Arbitrary Lagrangian-Eulerian (ALE) Method

Numerical algorithms using ALE method combine two classical kinematic descriptions of continuum (i.e. fluid and solid) mechanics; Lagrangian and Eulerian description. Lagrangian algorithms, in which each nodal point in the computational domain follow the movement of associated structures, are mainly used in solid mechanics. In contrast, Eulerian algorithms allow the continuum to move with respect to the fixed computational grids, thus widely used in fluid mechanics. By combining both algorithms, the computational grid can follow the moving objects in a Lagrangian way, while the fluid is still seen in an Eulerian manner (Donea, Huerta, Ponthot, and Rodríguez-Ferran, 2004).

ALE method was originally introduced in finite difference formulation (Hirt, Amsden, and Cook, 1974), and has been successfully implemented in finite volume and finite element formulations. Guardone, Isola, and Quaranta (2011) discretized the ALE formulation of Euler equations with finite volume method. They adopted edge-swap technique to improve the quality of triangular or tetrahedral cells in the deformation mesh and thus allow the boundaries to encounter large displacement. The technique is applied on translating and oscillating NACA 0012 airfoil case in which the mesh undergo large deformation.

Habchi, Russeil, Bougeard, Harion, Lemenand, Ghanem, Valle, and Peerhossaini (2013) developed a fluid-structure interaction solver using finite volume approach. Both governing fluid flow equation and structural displacement equation are discretized using finite volume method. ALE formulation is used to handle displacement of fluid-structure interfaces in the deforming mesh. Some benchmark two-dimensional problems such as lid-driven cavity with flexible bottom edge, elastic flap deformation induced by Von Karman vortex and two flaps in pulsating flow problem have been used to validate the solver. Through the combination of Pressure Implicit with Splitting of Operators (PISO) algorithm and Semi-implicit Method for Pressure-Linked Equations (SIMPLE) algorithm to solve fluid governing equation, they obtained accurate solutions by using large time steps.

Feistauer, Kučera, and Prokopová (2010) discretized the ALE form of compressible Euler equations using discontinuous Galerkin finite element method (DGFEM). Semi-implicit time discretization is used to avoid CFL-stability constraint thus allowing large time step to be taken for low Mach number problem. Then, Feistauer, Hasnedlová-Prokopová, Horáček, Kosík, and Kučera (2013) extended the method for viscous flow problems. They employed DGFEM as the discretization technique on ALE form of compressible Navier-Stokes equations. They showed that based on the validation of several numerical tests, the method can be applied to the fluid flow problem involving elastic structures.

Sun, Zhang, and Ren (2012) improved the characteristic-based split (CBS) scheme in ALE framework by reformulating the previous ALE-CBS scheme. Standard Galerkin finite element method is used for spatial discretization of governing flow equation while spring analogy method proposed by Blom (2000) is used in the moving mesh strategies. The improved ALE-CBS scheme is applied to the broken dam problem and the flow around oscillating circular cylinder. They found that the improved ALE-CBS scheme demonstrates better accuracy even using coarse mesh with large time step and is unaffected by mesh velocities in contrast to the former existing ALE-CBS scheme.

Implementation of ALE method requires re-meshing formulation to update the computational grids/meshes at each time step while at the same time avoiding severe mesh distortion and mesh entanglement. Due to the influence of re-meshing technique in the stability and accuracy of ALE method, many researches have enforced Geometric Conservation Law (GCL) when using this method.

The concept of GCL was derived by Thomas and Lombard (1979) in order to resolve the difficulties with maintaining global and local volume conservation due to the boundary-conforming coordinate transformation applied on flow computation involving moving boundaries. During such computation, the magnitude of transformation Jacobian (used to map variables in Cartesian coordinate system to a boundary-conforming curvilinear coordinate system) changes as the geometries of the boundary change in time. They found that, unsatisfying conservation of local volume due to such changes leads to erroneous solution. Therefore, GCL was addressed as a way to govern the changes so that it will not violate the local volume conservation. The concept was further investigated by (Guillard and Farhat, 2000) for solving time-dependent governing equations on dynamic mesh. They then introduced Discrete GCL (DGCL) as a useful guideline to evaluate geometric quantities involving grid positions and velocities. The law states the evaluation of such quantities should be conducted in a way that the numerical scheme used for integrating the flow equations must preserve a uniform flow field, independently of the mesh movement.

Since then, many researchers have studied the impact of GCL on solution accuracy. Thomas and Lombard (1979) implemented the GCL for density-based finite difference schemes on structured grids while Shyy, Udaykumar, Rao, and Smith (1996) implemented the GCL for pressure-based finite volume schemes. Lesoinne and Farhat (1996) developed first order time accurate scheme preserving the GCL using density-based ALE finite volume and finite element schemes on unstructured grids while Koobus and Farhat (1999) introduced second-order time accurate density-based

ALE finite volume schemes. Most studies show that not satisfying the GCL leads to wrong solutions or spurious oscillations in the solutions.

Mihara, Matsuno, and Satofuka (1999) presented an iterative finite-volume approach for unsteady coupled system of fluid and body motion in compressible flows. Similar to the dual-time or pseudo-time concept, inner iteration was carried out at every time step in order to satisfy the geometric conservation laws and to ensure accuracy. Gun-tunnel simulation was carried out using the method and based on the results, the method always satisfies the conservation laws independent of the CFL condition. The approach was further improved by using solution-adaptive moving-grid method (Sato, Matsuno, Nakagawa, and Satofuka, 2001). The improved version was validated by comparing numerical solutions of cylindrical implosion problem. It was proven to be more accurate than the solutions of uniform grid. On the other hand, Yamakawa and Matsuno (2004) presented the iterative finite volume method that includes algorithm for eliminating and merging cells of unstructured moving grids. The new method was developed for compressible flows and it was applied to a gun tunnel problem and two bodies docking and separating in a supersonic flow problem.

Visbal and Gordnier (2000) extended the HOC schemes for the solution of Navier-Stokes equations on moving grids. The authors used up to sixth order accurate Pade-type compact finite difference scheme combined with low-pass filtering technique while carried out time-marching using explicit (fourth order Runge-Kutta) and implicit (Newton-like sub iterative Beam-Warming scheme) time-integration method. Transformation Jacobian is evaluated at each time step using GCL in order to ensure free stream preservation. The extended scheme is applied on two and three-dimensional deforming 'wavy' mesh to investigate its accuracy. The performance of this scheme was also demonstrated by the simulation of viscous flow past a rapidly pitching NACA 0012 airfoil and the simulation of aeroelastic interaction arising from viscous flow past over a flexible panel. The simulation of pitching NACA 0012 airfoil was carried out using rotating (the grid rotates as the airfoil pitched at some angle) grid and deforming (the grids near airfoil boundaries distorted as the airfoil pitched at some angle) grid. Good agreement of both results indicated the robustness and versatility of this method. Furthermore, the resulting pressure field due to interaction of boundary layer and flexible panel exhibit acoustic radiation while the same phenomena does not arise in non-interacting case, thus showing the importance of computing fluid-structure interaction to capture the real physics behind such phenomena.

Kamakoti and Shyy (2003) state that GCL have been proven to be a key component of Computational Aeroelasticity problems specifically involving deforming grids. They developed a computational procedure for performing three-dimensional aeroelastic computations in a turbulent flow. Semi-implicit Method for Pressure-Linked Equations (SIMPLE) algorithm is used to solve the flow field, while the structure of deforming body is modeled by finite element. Multi-block structured grid with moving grid capability based on master/slave concept and transfinite interpolation concept was applied on the flow field while GCL is used to satisfy the conservation of discrete volumes. Simulation of AGARD 445.6 wing configuration in turbulent flows was performed to validate their method and it was shown that the results of aerodynamic parameters agree with the theory.

2.4.2 Immersed Boundary (IB) Method

Immersed boundary (IB) methods is a class of methods that simulate flows on computational grids that do not conform to the object's (or obstacle's) boundaries.

Originally developed by Peskin (1977) to simulate cardiac mechanics associated with blood flow, the method has gained popularity in solving flow problems with moving boundaries due to the use of stationary, non deforming Cartesian grid. IB is represented as a field of force in flow computation, unlike body-conformal grid, imposing boundary condition is not straightforward. Boundary condition is imposed indirectly through modification of governing equation by introducing source term (or forcing function) that will produce the effect of IB. Two kinds of approaches are used in that modification; continuous forcing and discrete forcing approach. The first approach is well suited with immersed elastic boundaries but posed accuracy and stability problems with immersed rigid boundaries, thus widely applied in biological studies where elastic boundaries abound. The second approach is much more difficult in terms of implementation of moving boundaries and require large computation grids for flow with high Reynolds number but enables greater accuracy near IB (Mittal and Iaccarino, 2005).

IB method has been widely used for incompressible flow simulation due to its advantages on elastic boundaries and low Reynolds number computation. Kim (2001) introduced the IB method based on a finite volume approach on a staggered grid and solved using fractional-step method. They introduced mass source and discrete-time momentum forcing into continuity and momentum equation of incompressible viscous flow, respectively. Mass source is applied at cell-center, while momentum forcing is applied in a staggered fashion on immersed boundaries or inside the body to satisfy mass continuity and no-slip boundary condition. It is shown that with the mass source included near immersed boundary, nonphysical solution especially near stagnation points is avoided and deterioration of numerical solution as Reynolds number increase is suppressed. The method has been further developed by Kim and Choi (2006) using

REFERENCES

Anderson, J. D. (1995). *Computational Fluid Dynamics: The Basics with Applications* (International edn.). New York: McGraw-Hill.

Arienti, M., Hung, P., Morano, E. & Shepherd, J. (2003). A level set approach to Eulerian-Lagrangian coupling. *Journal of Computational Physics 185* (1), 213-251.

Badr, H. M., Coutanceau, M., Dennis S. C. R., & Menard, C. (1990). Unsteady flow past a rotating circular cylinder at Reynolds numbers 10^3 and 10^4. *Journal of Fluid Mechanics 220*, 459-84.

Blom, F. J. (2000). Considerations on the spring analogy. *International Journal for Numerical Methods in Fluids* 32 (6), 647-668.

Čada, M. & Torrilhon, M. (2009). Compact third-order limiter functions for finite volume methods. *Journal of Computational Physics 228* (11), 4118-4145.

Chorin, A. J. (1967). A numerical method for solving incompressible viscous flow problems. *Journal of Computational Physics* 2 (1), 12-26.

Chung, T. J. (1997a). *Flowfield-dependent mixed explicit-implicit (FDMEI) algorithm toward direct numerical simulation in high speed flows* (Report No. F49620-95-1-0006).

Chung, T. J. (1997b). A new computational approach with flowfield-dependent implicitness algorithm for applications to supersonic combustion. In G. D. Roy, S. M. Frolov & P. Givi (eds), *Advance Computational and Analysis of Combustion* (pp. 466-489). Moscow: ENAS Publishers.

Chung, T. J. (1999). Transitions and interactions of inviscid/ viscous, compressible/ incompressible and laminar/ turbulent flows. *International Journal for Numerical Methods in Fluids 31* (1), 223-246.

Chung, T. J. (2002). *Computational Fluid Dynamics*. United Kingdom: Cambridge University Press.

Corcorant, A. J. (2010). *The Trouble With Zero*. Sydney: Palo Pacific Technology Pty Ltd.

Courant, R., Friedrichs, K., & Lewy, H. (1967). On the partial difference equations of mathematical physics. *IBM Journal of Research and Development 11* (2), 215-234.

Darwish, M. S., & Moukalled, F. (2003). TVD schemes for unstructured grids. *International Journal of Heat and Mass Transfer 46* (4), 599-611.

Deng, X. & Maekawa, H. (1997). Compact high-order accurate nonlinear schemes. *Journal of Computational Physics 130* (1), 77-91.

Diskin, B., Thomas, J. L., Nielsen, E. J., Nishikawa, H., & White, J. A. (2010). Comparison of Node-Centered and Cell-Centered Unstructured Finite-Volume Discretizations: Viscous Fluxes. *AIAA Journal 48* (7), 1326–1338.

Donea, J., Huerta, A., Ponthot, J. P., & Rodríguez-Ferran, A. (2004). Arbitrary Lagrangian–Eulerian Methods. In *Encyclopedia of Computational Mechanics* (Vol. 1. Chapter 14). John Wiley and Sons.

Elfaghi, M. A., Asrar, W., & Omar, A. A. (2009a). Accurate compact flowfield-dependent variation method for compressible euler equations. Paper presented at International Conference on Applications and Design in Mechanical Engineering, Penang.

Elfaghi, A. M., Asrar, W., & Omar A. A. (2009b). Comparison of High-order Accurate Schemes for Solving the Nonlinear Viscous Burgers Equation. *Australian Journal of Basic and Applied Sciences 3* (3), 2536-2543.

Elfaghi, A. M., Asrar, W., & Omar A. A. (2010). Higher order compact-flowfield dependent variation (HOC-FDV) solution of one-dimensional problems. *Engineering Applications of Computational Fluid Mechanics 4* (3), 434-440.

Feistauer, M., Kučera, V., & Prokopová, J. (2010). Discontinuous Galerkin solution of compressible flow in time-dependent domains. *Mathematics and Computers in Simulation 80* (8), 1612-1623.

Feistauer, M., Hasnedlová-Prokopová, J., Horáček, J., Kosík, A., & Kučera, V. (2013). DGFEM for dynamical systems describing interaction of compressible fluid and structures. *Journal of Computational and Applied Mathematics 254*, 17-30.

Forrer, H. & Berger, M. (1999). Flow simulations on Cartesian grids involving complex moving geometries. *International Series of Numerical Mathematics 129*, 315-324.

Garrison, T. J., Settles, G. S., & Hortsman, C. C. (1996). Measurements of the triple shock wave/turbulent boundary layer interaction. *AIAA Journal* 34 (1), 57-64.

Grétarsson, J. T., Kwatra, N. & Fedkiw, R. (2011). Numerically stable fluid-structure interactions between compressible flow and solid structures. *Journal of Computational Physics 230* (8), 3062-3084.

Guardone, A., Isola, D., & Quaranta, G. (2011). Arbirary Lagrangian Eulerian formulation for two-dimensional flows using dynamic meshes with edge swapping. *Journal of Computational Physics 230* (20), 7706-7722.

Guillard, H. & Farhat, C. (2000). On the significance of the geometric conservation law for flow computations on moving meshes. *Computer Methods in Applied Mechanics and Engineering 190* (11), 1467-1482.

Guillard, H., & Viozat, C. (1999). On the behaviour of upwind schemes in the low Mach number limit. *Computers & Fluids 28* (1), 63-86.

Habchi, C., Russeil, S., Bougeard, D., Harion, J. L., Lemenand, T., Ghanem, A., Valle, D. D., & Peerhossaini, H. (2013). Partitioned solver for strongly coupled fluid-structure interaction. *Computers & Fluids 71*, 306-319.

Harlow, F. H. & Welch, J. E. (1965). Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. *Physics of Fluid 8* (12), 2182-2189.

Heard, G. W. (2007). Flowfield-Dependent Variation (FDV) Method for Compressible, Incompressible, Viscous, and Inviscid Flow Interactions with FDV Adaptive Mesh Refinements and Parallel Processing. Unpublished doctoral dissertation, University of Alabama, Huntsville.

Hirsch, C. (1990). *Numerical computation of internal and external flows volume 2: computational methods for inviscid and viscous flows*. Chichester: John Wiley & Sons.

Hirsh, R. S. (1975). Higher order accurate difference solutions of fluid mechanics problems by compact differencing technique. *Journal of Computational Physics 19*, 90-109.

Hirt, C. W., Amsden, A. A., & Cook, J. L. (1974). An arbitrary Lagrangian–Eulerian computing method for all flow speeds. *Journal of Computational Physics 14* (3), 227-253.

Hoffmann, K. A. & Chiang, S. T. (2000). *Computational Fluid Dynamics Volume I* (4th edn.). Wichita: Engineering Education System.

Hu, F. Q., Li, X. D., & Lin, D. K. (2008). Absorbing boundary conditions for nonlinear Euler and Navier–Stokes equations based on the perfectly matched layer technique. *Journal of Computational Physics 227* (9), 4398-4424.

Kamakoti, R. & Shyy, W. (2003). Moving grid computations for turbulent, aeroelastic flows. Paper presented at 33rd AIAA Fluid Dynamics Conference and Exhibit, Florida.

Kirshman, D. J., & Liu, F. (2006). Flutter prediction by an Euler method on non-moving Cartesian grids with gridless boundary conditions. *Computers & Fluids 35*, 571-586.

Kim, D. (2009). Laminar flow past a sphere rotating in the transverse direction. *Journal of Mechanical Science and Technology 23*, 578-589.

Kim, D. & Choi, H. (2006). Immersed boundary method for flow around an arbitrarily moving body. *Journal of Computational Physics 212* (2), 662-680.

Kim, J. (2001). An immersed-boundary finite-volume method for simulations of flow in complex geometries. *Journal of Computational Physics 171* (1), 132-150.

Kim, Y. & Peskin, C. S. (2009). 3-d parachute simulation by the immersed boundary method. *Computers & Fluids 38* (6), 1080-1090.

Landon, R. H. (1982). *Compendium of unsteady aerodynamic measurements: Data set 3, NACA 0012 Oscillatory and transient pitching* (Report No. AGARD-R-702). Advisory Group for Aerospace Research and Development.

Lele, S. (1992). Compact finite difference schemes with spectral-like resolution. *Journal of Computational Physics 103*, 16-42.

Lesoinne, M. & Farhat, C. (1996). Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations. *Computer Methods in Applied Mechanics and Engineering 134* (1), 71-90.

Lomtev, I., Kirby, R. M., & Karniadakis, G. E. (1999). A discontinuous Galerkin ALE method for compressible viscous flows in moving domains. *Journal of Computational Physics 155* (1), 128-159.

Mawlood, M. K., Basri, S., Asrar, W., Omar, A. A., Mokhtar, A. S., & Ahmad, M. M. (2006). Solution of navier-stokes equations by fourth-order compact schemes and AUSM flux splitting. *International Journal of Numerical Methods for Heat and Fluid Flow 16* (1), 107-120.

Megahed, A. A., El-Mallah, M. W., & Girgise, B. R. (2006). A modified flowfield dependent variation method applied to the compressible euler flow equations. Paper presented at 8th International Congress of Fluid Dynamics and Propulsion, Sinai.

Mihara, K., Matsuno, K., & Satofuka, N. (1999). An iterative finite-volume scheme on a moving grid: 1st report, the fundamental formulation and validation. *Transactions of the Japan Society of Mechanical Engineers Series B 65* (637), 2945-2953.

Mittal, R., Dong, H., Bozkurttas, M., Najjar, F. M., Vargas, A., & Von Loebbecke, A. (2008). A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries. *Journal of Computational Physics 227* (10), 4825-4852.

Mittal, R. & Iaccarino, G. (2005). Immersed boundary method. *Annual Review of Fluid Mechanics 37*, 239-261.

Mohaghegh, M. R., & Jafarian, M. M. (2010). Unsteady Transonic Aerodynamic Analysis for Oscillatory Airfoils using Time Spectral Method. *International Journal of Mechanical and Materials Engineering 1* (4), 264-271.

Murman, S. M., Aftosmis, M. J., & Berger, M. J. (2003). Implicit approaches for moving boundaries in a 3-D Cartesian method. Paper presented at 41st AIAA Aerospace Sciences Meeting, Nevada.

Newmark, N. M. (1959). A method of computation for structural dynamics. *Journal of the Engineering Mechanics Division 85* (EM3), 67-94.

Nkonga, B. & Guillard, H. (1994). Godunov type method on non-structured meshes for three-dimensional moving boundary problems. *Computer Methods in Applied Mechanics and Engineering 113* (1), 183-204.

Nonomura, T., Iizuka, N., & Fujii, K. (2010). Freestream and vortex preservation properties of high-order WENO and WCNS on curvilinear grids. *Computers & Fluids 39* (2), 197-214.

Ou, K., Castonguay, P., & Jameson, A. (2011). Computational sports aerodynamics of a moving sphere: simulating a ping pong ball in free flight. Paper presented at 29th AIAA Applied Aerodynamics Conference, Hawaii.

Paz, M. & Leigh, W. (2006). *Structural Dynamics: Theory and Computation* (5^{th} edn.). Massachusetts: Kluwer Academic Publishers.

Paillére, H., Clerc, S., Viozat, C., Toumi, I., & Magnaud, J. P. (1998). Numerical methods for low mach number thermal-hydraulic flows. Paper presented at 4th European Computational Fluid Dynamics Conference, Greece.

Persson, P. O., & Peraire, J. (2008). Newton-GMRES preconditioning for discontinuous Galerkin discretizations of the Navier-Stokes equations. *SIAM Journal on Scientific Computing 30* (6), 2709-2733.

Peskin, C. S. (1977). Numerical analysis of blood flow in the heart. *Journal of Computational Physics 25* (3), 220-252.

Ramamurti, R., Sandberg, W., Löhner, R., Walker, J., & Westneat, M. (2002). Fluid dynamics of flapping aquatic flight in the bird wrasse: three-dimensional unsteady computations with fin deformation. *Journal of Experimental Biology 205* (19), 2997-3008.

Richardson, G. A. & Chung, T. J. (2002a). Computational relativistic astrophysics using the flow field-dependent variation theory. *The Astrophysical Journal Supplement Series 139*, 539-563.

Richardson, G. A. & Chung, T. J. (2002b). General relativistic MHD with the flowfield dependent variation method. Paper presented at American Physical Society, April Meeting, jointly sponsored with the High Energy Astrophysics Division (HEAD) of the American Astronomical Society, New Mexico.

Richardson, G. A., Chung, T. J., Karr, G. R., & Pendleton, G. N. (2000). Flowfield dependent variation method for complex relativistic fluids. Paper presented at Gamma-Ray Bursts: 5th Huntsville Symposium, Alabama.

Richardson, G. A., Cassibry, J. T., Chung, T. J., & Wu, S. T. (2010). Finite element form of FDV for widely varying flowfields. *Journal of Computational Physics 229* (1), 145-167.

Saad, Y., & Schultz, M. H. (1986). GMRES: A generalized minimal residual algorithm for solving nonsymmetric linear systems. *SIAM Journal on Scientific and Statistical Computing* 7 (3), 856-869.

Sato, Y., Matsuno, K., Nakagawa, Y., & Satofuka, N. (2001). An iterative finite-volume scheme on a moving grid: 2nd report, construction of a solution-adaptively moving-grid method for unsteady compressible flows. *Transactions of the Japan Society of Mechanical Engineers Series B 67* (653), 23-28.

Schunk, G., Canabal, F., Heard, G., & Chung, T. J. (1999). Unified CFD methods via flowfield-dependent variation theory. Paper presented at 30th AIAA Fluid Dynamics Conference, Virginia.

Schneiders, L., Hartmann, D., Meinke, M., & Schröder, W. (2013). An accurate moving boundary formulation in cut-cell methods. *Journal of Computational Physics 235*, 786-809.

Sen, S., Kalita, J. C., & Gupta, M. M. (2013). A robust implicit compact scheme for two-dimensional unsteady flows with a biharmonic stream function formulation. *Computers & Fluids 84*, 141-163.

Shyy, W., Udaykumar, H. S., Rao, M. M., & Smith, R. W. (1996). *Computational Fluid Dynamics with Moving Boundaries*. Washington DC: Taylor and Francis.

Sod, G. A. (1978). A survey of several finite difference methods for systems of nonlinear hyperbolic conservation laws. *Journal of Computational Physics 27* (1), 1-31.

Sun, X., Zhang, J. Z., & Ren, X. L. (2012). Characteristic-based split (CBS) finite element method for incompressible viscous flow with moving boundaries. *Engineering Applications of Computational Fluid Mechanics 6* (3), 461-474.

Thomas, P. D., & Lombard, C. K. (1979). Geometric conservation law and its application to flow computations on moving grids. *AIAA Journal 17* (10), 1030-1037.

Turkel, E., Fiterman, A., & Van Leer, B. (1993). *Preconditioning and the limit to the incompressible flow equations* (Report No. ICASE-93-42). Institute for Computer Applications in Science and Engineering.

Van Leer, B. (1979). Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunov's method. *Journal of Computational Physics 32* (1), 101-136.

Venkatakrishnan, V., & Mavriplis, D. J. (1996). Implicit method for the computation of unsteady flows on unstructured grids. *Journal of Computational Physics 127* (2), 380-397.

Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method* (2nd edn.). Harlow: Pearson Education Limited.

Visbal, M. & Gordnier, R. (2000). A high-order flow solver for deforming and moving meshes. Paper presented at Fluids 2000 Conference and Exhibit, Denver.

Visbal, M. R., & Shang, J. S. (1989). Investigation of the flow structure around a rapidly pitching airfoil. *AIAA journal 27* (8), 1044-1051.

Weiss, J. M. & Smith, W. A. (1995). Preconditioning applied to variable and constant density flows. *AIAA Journal 33* (11), 2050-2057.

Wesseling, W. (2000). *Principles of Computational Fluid Dynamics* (1st edn.). Berlin: Springer.

Woodward, P. & Colella, P. (1984). The numerical simulation of two-dimensional fluid flow with strong shocks. *Journal of Computational Physics 54* (1), 115-173.

Xia, G., & Lin, C. L. (2008). An unstructured finite volume approach for structural dynamics in response to fluid motions. *Computers and Structures 86* (7), 684-701.

Yamakawa, M. & Matsuno, K. (2004). An iterative finite-volume method on an un-structured moving grid: 2nd report, application to moving boundary problem with body separation/mergence. *Transactions of the Japan Society of Mechanical Engineers Series B 70* (689), 25-30.

Yoon, K. T., & Chung, T. J. (1996). Three-dimensional Mixed Explicit-implicit Generalized Galerkin Spectral Element Methods for High-speed Turbulent Compressible Flows. *Computer Methods in Applied Mechanics and Engineering 135* (3), 343-367.

Yoon, K. T., Moon, S. Y., Garcia, S. A., Heard, G. W. & Chung, T. J. (1998). Flowfield-dependent Mixed Explicit-implicit (FDMEI) Methods for High and Low Speed and Compressible and Incompressible Flows. *Computer Methods in Applied Mechanics and Engineering 151* (1), 75-104.

Yu, M. L., Wang, Z. J., & Hu, H. (2011). A high-order spectral difference method for unstructured dynamic grids. *Computers & Fluids 48* (1), 84-97.